

## CFD ANALYSIS OF FLOW OVER BLUFF BODIES WITH VARIOUS RANGES OF INJECTION USING ANSYS CFX

R. NALLAPPAN<sup>1</sup>, M. SUNDARARAJ<sup>2</sup> & A. ADAIKALARAJ<sup>1</sup>

<sup>1</sup>Assistant Professor, Department of Aeronautical Engineering, Bharath Institute of Higher  
Education and Research, Tamil Nadu, India

<sup>2</sup>Head and Associate Professor, Department of Aeronautical Engineering, Bharath Institute of Higher  
Education and Research, Tamil Nadu, India

### ABSTRACT

*This paper presents the computational analysis of the effect of injection techniques on bluff bodies. In this present study, a blunt nosed model was designed with and without injections at the frontal area. Three different injection configurations were chosen for our study (0.5, 1, and 1.5). The analyses were further extended by changing the velocities. The velocities considered for this study is 50m/s, 70m/s and 80m/s. Efforts were taken to obtain the flow pattern and the pressure distribution zone over the nose of the blunt body. Flow pattern was observed and recorded for different injection configurations. A comparative study was also done to observe the better pressure zone. It is observed that the radius of curvature of the streamline gets changed due to injection. Also, there occurs a deflection in the streamline. By this study it is concluded that the flow pattern and the positive pressure zone gets affected by the injection technique.*

**KEYWORD:** Injection, Pressure Zone, Blunt-Nose Body & Vortex

**Received:** Mar 14, 2019; **Accepted:** Apr 04, 2019; **Published:** May 03, 2019; **Paper Id.:** IJMPERDJUN201967

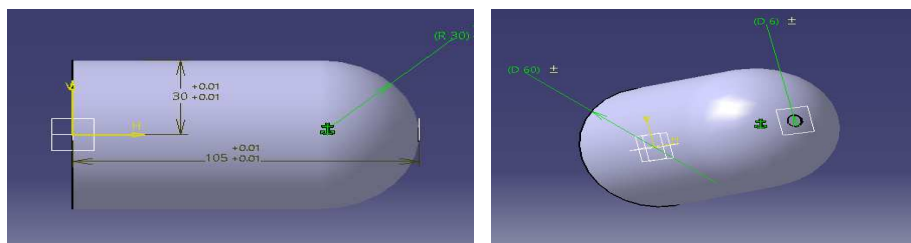
### INTRODUCTION

In low speed and high speed applications, the drag reduction in blunt-nose body plays an important role. The formation of high drag in hypersonic flying objects is because of the blunt nose region. This blunt region leads to high pressure zone at the front region of nose. But we could not change the blunt shape of the vehicles since this shaped nose acts as the safeguard of the body during aerodynamic heating condition. Changes can be made in the frontal region by blowing inside in terms of injection. Secondary designs were made in front of the blunt nose region projecting inside to reduce drag. This is accepted as the effective method of drag reduction. These injections actually shift the forward stagnation point away from the nose region. Because of this change in the location of stagnation point, the extent of the pressure zone comes down which in turn resulted in drag reduction. The flow process related to the shifting of the stagnation point in front of the blunt nose will result in reducing the strength of the detached shock, and reduces the extension of the pressure zone influenced by the detached shock. Many experimental studies have been conducted to analyze the frontal body flow field of the blunt body with injections. This present work is an attempt to measure the positive zone of influence, and the pressure field over the frontal region of a blunt-nosed body, with and without injections. To study this, the flow pattern over the desired blunt-nosed body, with without injections, was analyzed and visualized using ANSYS software packages, at a specific Reynolds number. The Computational analysis is one of the cheap and best techniques which can be used for the

visualization of real flow problems. William H. Bettes et al., presented a paper. In his paper, the drag coefficient on the automobiles can be reduced by 24% by changing the front body design shapes and 11% by changing rear body design of the vehicle which saves 25 percent fuels consumption. Edwin J. Saltzman et al. conducted experimental studies on blunt bodies. From their experiments, the rounded corner vehicle when compared with square shaped configurations shows that reduction in aerodynamic drag of about 40 percent can be achieved when compared to the square shaped configuration. Ashish Vashishtha, et al., made an attempt to study the comparative results between drag on the blunt nosed body with and without breathing nose at the Mach 1.96. The breathing nose resulted in less drag because of the manipulation of the high pressure at the nose and the low pressure at the base simultaneously. A maximum drag reduction of 21% was obtained. This drag reduction is because of the combined effect of decrease of high pressure at the nose and increase of low pressure at the base. In this work, we are considering the breathing technique in terms of injections to reduce the drag.

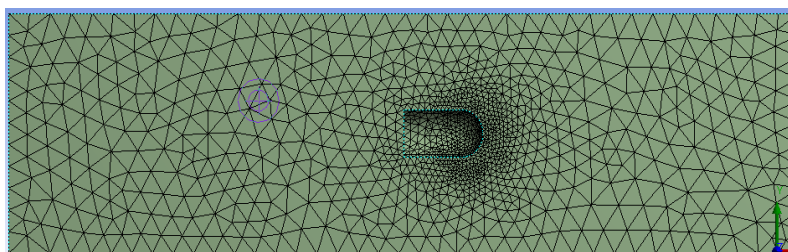
## MODELLING AND SIMULATIONS

The model was designed using CATIA V5 software. In the present study the flow around a blunt body without injections, blunt body with injection values of 0.5, 1 and 1.5 were designed and analyzed. The models were shown in figure 1. The overall length of the blunt body was considered as 90mm and the width of the blunt body was taken as 60mm. The nose is having a diameter of 60mm. The analysis were further extended by changing the velocities as 50m/s, 70m/s and 80m/s.



**Figure 1: Bluff Bodies with Injections**

Figure 2 shows the discretized model of the blunt bodies with injections. The domain was created around the model since analyses deals with external aerodynamics. The models designed using CATIA V5 were imported to ICEMCFD for meshing and the discretized model was shown below. Meshing was done finely near the areas where we need to capture the flow field. Unstructured mesh was selected for this meshing process. After the completion of discretization stage, the model was imported for analysis purposes.



**Figure 2: Discretized Model**

## RESULTS AND DISCUSSIONS

Four models were chosen for our analysis. Air is allowed to flow over the different models at different velocities. Boundary conditions were given and the analyses were carried out for four different configurations. The results were

observed and recorded in this paper. The obtained results were compared with the base shape without injections. In order to reduce the drag over these kind of blunt bodies, we need to reduce the positive pressure at the nose region. By placing injection in front of the nose region we could reduce the positive pressure field region over the nose area.

### Pressure Contours

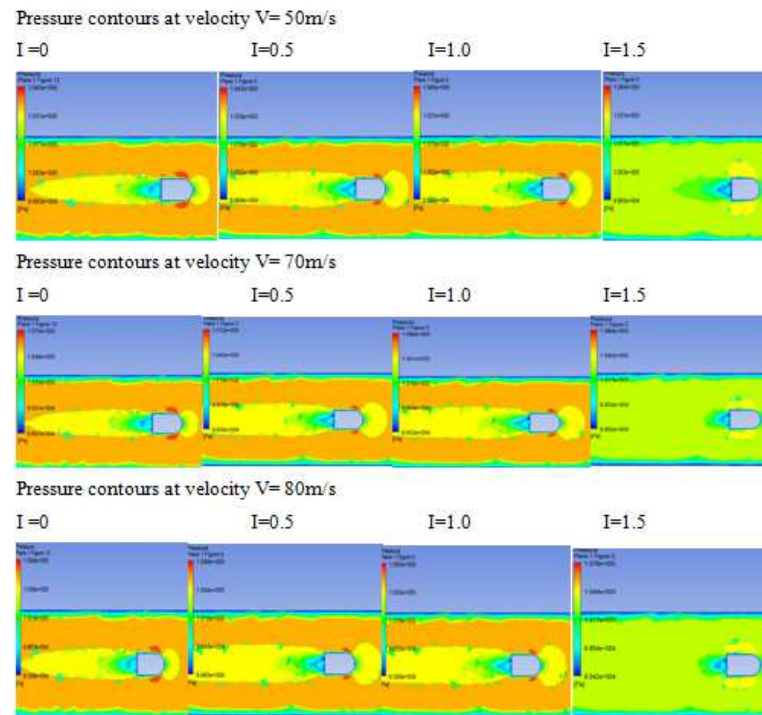


Figure 3: Pressure Contours for Various Configurations at Different Velocities

### Temperature Contours

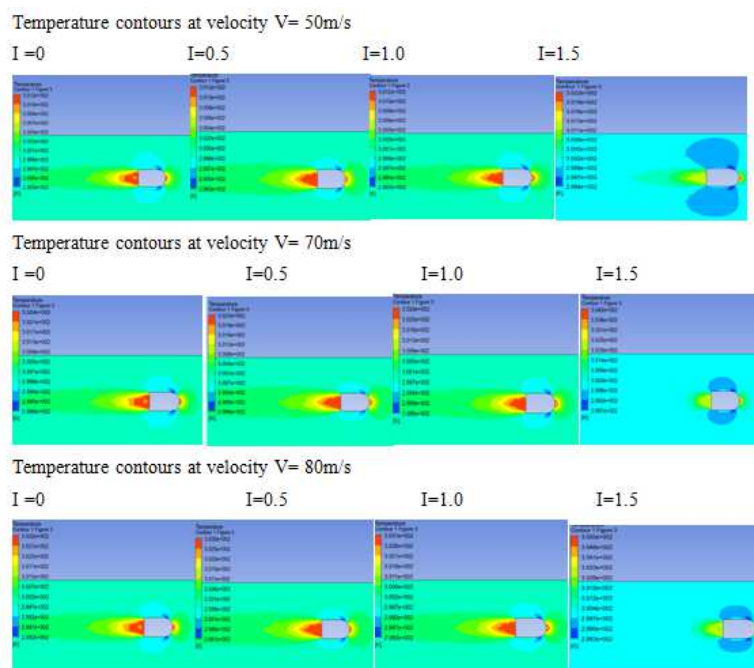
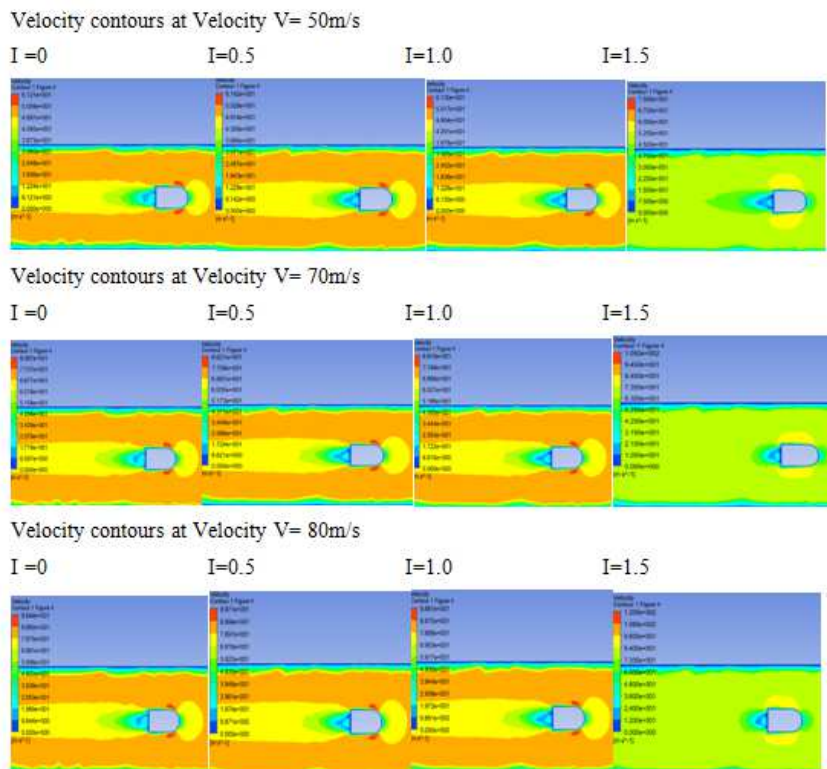


Figure 4: Temperature Contours for different Configurations at Various Velocities

## Velocity Contours



**Figure 5: Velocity Contours for different Configurations at Various Velocities**

## CONCLUSIONS

The injection configurations attached to the basic blunt nosed body have much influence on the flow field, twin vortex formation and reverse flow distance. There is a considerable increment in the reverse flow distance for model with and without injections. It is observed that the injected model has given the lowest vortex length compared to the basic blunt nosed body. Whereas coming to the three different configuration of blunt bodies, it is observed that the effect on the flow field is insignificant. It is found that the injection induced model reduces the radius of curvature of the approaching streamline, and the deflection of the streamline is towards the shoulder of the basic body resulting in a narrow zone of positive pressure hill at the nose. So finally it can be concluded that choosing perfect injection configuration will be beneficial as the economic concern.

## REFERENCES

1. William H. Bettes "The aerodynamic drag reduction of road vehicles in earlier" *Journal Of Engineering & Science* / January 1982
2. Edwin j, Saltman and Robert R. Meyer "drag reduction obtained by rounding vertical corners of box shaped vehicle" *NASA Flight research center* march 1974.
3. Ashish Vashishtha, Hemant Sharma, P. Lovaraju, and E. Rathakrishnan "Breathing Blunt Nose Concept for Drag Reduction In SupersonicFlow"
4. E. Rathakrishnan "Effect Of Splitter Plate On Bluff Body Drag" *Aiaa Journal* Vol. 37, No.9, September 1999

5. Khalid M. Sowed and E. Rathakrishnan "Front Body Effects On Drag And Flow Field Of A 3-D Bluff Body" *AIAA Journal*, VOL.31, NO 7;technical notes
6. G. K. Suryanarayana, Hemming Pauer, G. E. A. Meier "Bluff-Body Drag Reduction By Passive Ventilation" *Journal of Experiments in Fluids*16, 73-81 (1993).
7. Chaudhari, S. C., Yadav, C. O., & Damor, A. B. (2013). A comparative study of mix flow pump impeller cfd analysis and experimental data of submersible pump. *IJRET*) ISSN, 2321-8843.
8. Stephen A. Whitmore, Stephanie Sprague, Jonathan W. Naughton "Drag Reduction Using Fore Body Surface Roughness" NASA/TM-2001-210390
9. Shashank Khurana, Kojiro Suzuki , Ethirajan Rathakrishnan," Flow Field around a Blunt-nosed Body with Spike, *International Journal of Turbo & Jet-Engines*, Volume 29, Issue 4, Dec 2012, pp 217-221.
10. R. Kalimuthu and E. Rathakrishnan, Aerodynamic Characteristics of Blunt Nosed Body with and without Aerospike at Mach 5.87, *Proceedings of the 10th Asian Symposium on Visualization*, 2010
11. V. L. Zhdanov, H. D. Papenfuss "Bluff Body Drag Control By Boundary Layer Disturbances" *Journal of Experiments in Fluids* 34 (2003)460–466
12. Sang-Joon Lee, Sang-Ik Lee, Cheol-Woo Park "Reducing The Drag On A Circular Cylinder By Upstream Installation Of A Small Control Rod" *journal of Fluid Dynamics Research* 34 (2004) 233–250.
13. Sawant, S. M., & Hujare, D. P. (2013). Thermo-mechanical analysis for skirt of pressure vessel using FEA approach. *International Journal Of Mechanical Engineering (IJME)* ISSN, 2319-2240.
14. Yoshio Yajima, Osamu Sano "A note on the drag reduction of a circular cylinder due to double rows of holes" *Journal Of Fluid Dynamics Research* 18 (1996) 237-243.

